16

Simulation of Net Structures Hydrodynamic Fields

Leonid Meyler
The Baltic State Fishing Fleet Academy
Russia

1. Introduction

Important problems in industrial fishery are catchability and selectivity of fishing gears. These characteristics depend on various parameters of gears that usually have a highly hydroelastic net structure without a predefined shape. The shape forms in motion or under flow action. Curved net structures are usually highly three-dimensional. The study of the net structure volume deformation is complex, requiring treatment of structural properties like flexibility, discontinuity, anisotropy and uneven surface shapes. Hydrodynamics of fishing net structures, in particular, a fishing trawl and its rear part – cod-end, attracts an attention of scientists for a long time. Fig. 1 shows a trawl system scheme.

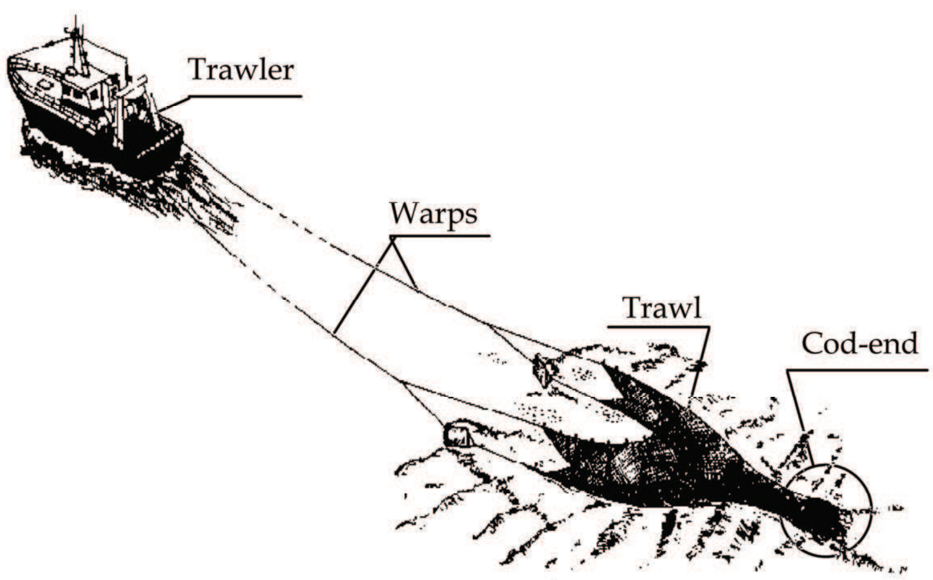


Figure 1. Scheme of a fishing trawl system

The dynamic interaction between a net structure, a fishing vessel, rigging, warps and other trawl elements connected together are complex. An importance of flow velocities and pressures estimation both around and in trawl consists in an exclusive dependence of the forces acting on each part of the trawl system. The knowledge of these forces allows to define the shape, drag and behavior of the structure during a trawling process, tension and loads in its twines and ropes. As well, flow velocities and pressures in the cod-end, have a very significant influence on the fish motion process. The quantity and quality of the catch collected in the cod-end depends on fish behavior. Besides influencing of the cod-end design on flow velocities/pressures it causes a net "meshing" with fish that in its turn influences hydrodynamics of the cod-end and consequently on its catchability and selectivity. From the point of view of modern fishing engineering (Paschen et al., 2001) the key to understand these characteristics is to complete consideration of fish, fishing gear and fluid as a whole. Therefore velocity and pressure distribution ahead of and in the fishing gear have to be predicted on a basis of experimental investigations and numerical simulations.

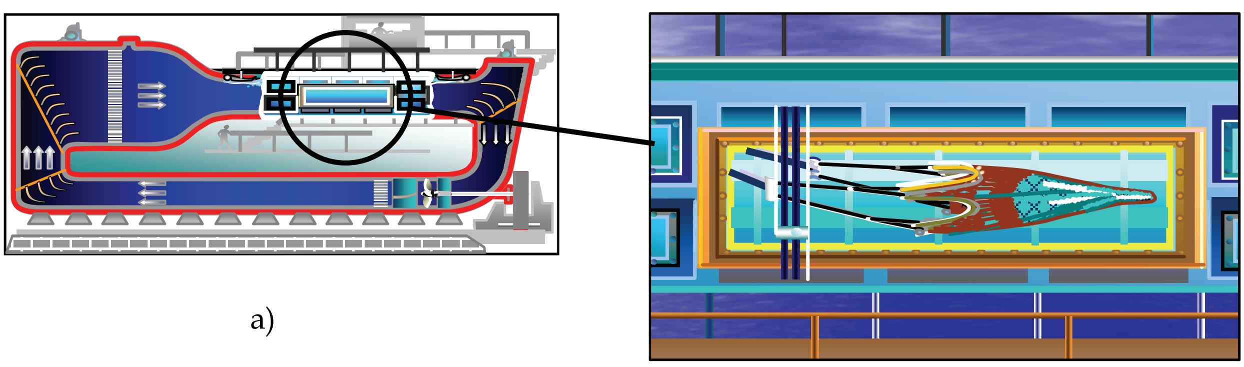
2. A brief review of previous studies

This article does not pursue the objective to give a general review of such works. Nevertheless, it is possible to mention some of them. The earliest publications described experimental methods of testing net cylinders, cones, models of trawls and real trawls. Results have been presented usually as graphic dependences of flow velocities or pressures on net structures characteristics and empirical expressions describing these dependences.

Cones as a main part of a trawl design were studied more often. Such investigations have been carried out using special experimental plants. For example, a flume tank was used (Higo, 1964) for velocities measurements around net cones with a thermister current meter and experimental data and empirical dependences of flow velocities on cones characteristics were presented. A research of different trawl models (Stengel & Fischer, 1964) conducted in a wind tunnel allowed to evaluate pressures distribution in front of and into the models. A Pitot tube used for these experiments. A wind tunnel also was used as the experimental plant for tests (Rehme & Scheel, 1988) of a big size net cone made of different size netting. Nondimensional velocities as a function of cross coordinates were resulted. A net cone made of metal wire was studied (Scharping, 1974) in a wind tunnel in order to measure pressures at the different longitudinal sections both in and outside the model. Cones towed by a vessel in order to investigate velocities influence at the net "mouth" on velocities inside the net and to get the correspondent dependence (Higo et al, 1974). Experiments (Imai, 1974) carried out in a circular tank permitted to study velocities distribution inside trawl models made of thin and thick net webbing.

The big contribution to the problem studying has been brought by Z. Ziembo (Ziembo, 1974; 1987 a, b; 1988; 1989; 1993). The author carried out many experimental researches and suggested mathematical models to describe a process "water flows in trawls". These articles describe methods of investigations, results and comparison between theoretical and experimental approaches. In particular, trawl models having different mesh sizes have been towed by a catamaran equipped with special facility for direct observations and measurements. Velocity and pressure distributions around the models have been compared both to a simplified mathematical model based on considering of ideal, incompressible non-viscous liquid and to mathematical model for real water. Another experiment devoted to study of kinetics of water flow through trawl models and real trawl. Solving this problem has a great importance since it gives possibility to design trawls with best shape parameters adapted to fish reactions on the net structure. Calculations of water flow velocity through the cod-end under the assumption that the flow is uniform and equal on its entire surface were made (Moderhak, 1993) for different ratios of codend length to its diameter, mesh bar length to mesh bar diameter, and for various mesh opening coefficients. The results obtained were analyzed with regard to the impact of flow velocity on hydrodynamic forces

opening the cod-end. A simplified mathematical model was used for flow velocities normal to a cod-end calculation. An idea of a new set-up of meshes in the codend, changing the effect of operation of longitudinal forces (hydromechanical drag forces) was presented.
Hydromechanics of towed trawl models was investigated (Gabriuk & Chernetsov, 1985) by special designed catamaran. Some of researchers (Kroeger, 1984; Ziembo, 1989) investigated towed full-scale trawls in order to evaluate dependences of longitudinal flow velocities along the different design trawls using propeller type flow meters. Others (Miziurkin & Kostiukov, 1982; Kostiukov & Shevchenko, 1983) studied fish behavior and hydrodynamic fields by means of special detectors put inside real trawls. Data registration made from an underwater manned vehicle.
Serial experimental researches on net cones and cylinders, trawl models, cod-end models with imitation of a catch have been carried out in one of the world largest flume tanks, which is located in Kaliningrad, Russia. Fig. 2 shows its scheme. Many publications followed these experiments. A trawl cod-end model was tested (Baev & Belov, 1987) for the velocity field determination. As a result of trawl cod-ends investigations (Korotkov & Meyler, 2000; 2001), an improved design of the cod-end was proposed and realized for industrial trawlers.



a) b)

Figure 2. View of the flume tank (a) and the working part (b)

Flow velocities have been measured in experiments (Belov, 1987, 2002) both inside and outside of net construction. An example of longitudinal flow velocities experimental data is shown in Fig. 3. Special attention is paid to the theoretical and experimental methods for the design of fishing gears and investigations on cod-end selectivity in the framework of the international workshops "Methods for the Development and Evaluation of Maritime Technologies - DEMaT". Measurements of the flow by using the Scanmar speed sensor were carried out (Thiele, 1997) inside and outside of a selectivity cod-end. The results show that the velocity of the flow inside the cod-end is strongly influenced by the mesh size and the cover. Systematic model experiments on net cones were carried out (Paschen & Winkel, 1999) in order to obtain the necessary quantitative relation between the known design of the net shape and the corresponding flow and pressure distributions in the environment as well as inside the net structure.

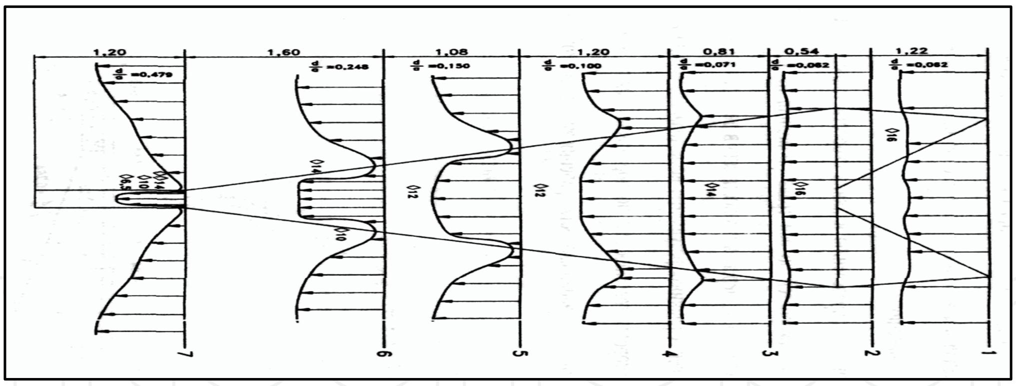


Figure 3. Flow speed distribution in/around a trawl model

The suggested methodology is suitable to carry out basic analysis of net-fluid interactions. Among the received experimental results, the geometrical angle of attack of the net cone is of crucial importance for the velocity and pressure distributions already in front and within the reticulate structure. In addition, the paper gives an approach to approximate calculation of pressure increase in a towed conical net using assumption that meshes themselves and the caught fish cause a blockage effect and drag. This influence is of primary importance for static pressure and velocity inside the cone. Authors notice that the structure fluid interactions with trawls and their elements are very complex due to the high flexibility of net constructions and mathematical modeling of this interactions become more and more important in connection with the efforts to increase the selectivity of trawls. Considering methods and limits for wind tunnel tests (Winkel & Paschen, 2001) an importance of the analysis of temporally variable flows inside or around fishing nets was emphasized. It gives a better understanding of fish behaviour and net selectivity. Flow velocities was measured (Enerhaug, 2005) in conical shaped models with high net solidity. Experiments have been carried out in a flume tank in order to verify theoretical approaches for modeling, predicting and interpreting the flow in fine-meshed pelagic trawls. An experimental approach for the analysis of the flow distribution in a brailer-codend (Fuwa et al., 2007) was studied in a flume tank. Flow speed inside the codend was measured using the current meter.

According to a three-dimensional numerical model of the fluid flow (Fredheim & Faltinsen, 2001), that takes into consideration the influence of the structure on the flow, the net is divided into a set of cylindrical and spherical elements to represent the twines and knots respectively. Each of these different elements is modeled as a set of source distribution and single point sources. This model allows to describe the disturbance of the fluid flow due to the presence of the net and to determine the influence from the net on the flow.

The study (Benoit & Marichal, 1996) took place in a global improvement project of trawl selectivity, which means its capacity to let young fish escape through the meshes. The authors used Navier-Stokes equations for calculation of velocities in the cod-end. They undertake necessity to know water velocity in the trawl to be able to calculate mesh geometry. Indeed, mesh geometry depends on hydrodynamic forces due to flow. The cod-end is supposed to be homogeneous and permeable surface, because equations mentioned above cannot describe flow around each twine of cod-end. Specific equations were used to compute tangential and normal components at the net's surface.

In spite of the significant experimental and theoretical material on the problem which has been collected since sixties of the last century, recent publications devoted to increasing of reliability of flow parameters in net structures and simulation methods development

testifies that the problem remains actual at present time. Interesting researches (O'Neill, 2003, Priour & Herrmann, 2005) are devoted to factors influencing the cod-end shape under a flow and with caught fish.
As it was said in the above mentioned study (Paschen et al., 2001), the predictability of the flow field inside and outside a net structure is necessary in order to simulate the connection between net structure, fluid and fish. Authors underline that even if highly efficient Computational Fluid Dynamics (CFD) methods are applied, the problem seems to be rather complex. A study (Enerhaug at al., 2003) gives experimental tests description of the flow velocity field around a solid cone and a reticulate one in the flume tank. Then experimental data were compared with numerical calculations. The flow depends on the cone solidity value. Authors used FEMLAB CFD software for the two-dimensional simulation where the reticulate cones were idealized by a number of slender parallel cylinders, authors made a conclusion that such approach only meant to illustrate the overall physical principles. However, such simulations give the basis for further research because the real net structure has a dynamic moving, equipped with rigging elements, etc. Anyway, if results of experiments and calculations will have a rather good convergence, this method can be used for hydrodynamic field characteristics prediction. Due to the fact that netting can be approximated by very large number of small cylinders connected in knots, an approach (Patursson at al., 2007) to simulate the net as a thin volume of porous media was presented. Fig. 4 shows a scheme of the suggested method.

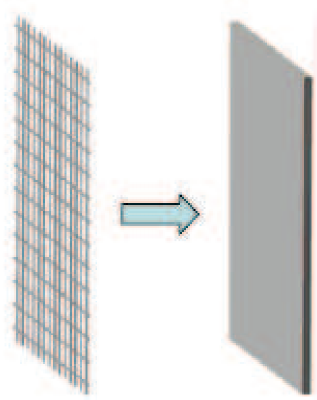


Figure 4. The net conversion to a thin volume of porous media

Predicted drag- and lift coefficient of the net panel and the velocity reduction behind the net panel was calculated by the model available in FLUENT CFD software. These data were compared against values measured during experiments in a flume tank. The modeled results have a good agreement with measured values.

3. CFD software description

As it was mentioned above, more and more researches have now numerical character when the various software products based on the equations of mathematical physics decision are used. Methods of computer modelling give unique technologies of physical fields studies. It is possible to use ready programs and to model various combinations of the interconnected physical fields. Thus the engineer, the designer or the researcher does not have necessity to create programs for the decision of the problems. One of the applications is hydrodynamics of various bodies. For example, it can be a combination of velocity or pressure fields inside and around net structures. Certainly, at such statement it is important to be sure that the software is based on a mathematical model adequate to the real physical process. Thus, reliability of numerical calculation has to be supported by comparison with experimental data that will allow to evaluate the so-called «scale effect». Development of mathematical models and the software are based on CFD methods. CFD is a modern component of market CAE (Computer Aided Engineering). CFD is based on FEM (the Finite Element Method) or FEA (the Finite Element Analysis). This article has no purpose of making a thorough survey of similar software products, but some of them will be mentioned as an opportunity for the concrete applied problem decision - a numerical estimation of a net structures hydrodynamic field.

The ANSYS software (http://www.ansys.com) is the program of the finite-element analysis. ANSYS gives a possibility to apply computing methods of hydrodynamics for parameters of a liquid or gas flow definition. The solved problems can be stationary or non-stationary and the flow can include up to six components disconnected among themselves. The hydrodynamic analysis is used for definition of such parameters of the liquid as a pressure difference, a velocities distribution, a flow direction, lift and drag forces, influence of heating and cooling. Velocity components, values of pressure and temperature are defined on the basis of laws of mass preservation, quantity of motion and energy. For turbulent motion modelling there is an opportunity to use the description of the phenomenon by means of the continuity and impulse equations.

Another CFD package FLOW-3D (http://www.flow3d.com) is capable to model the big variety of problems of liquid flows. They can be flows with a free surface, and the limited and internal flows. It is expressed in the simplified approach to generation of a finite-element grid that affects an accuracy increase, reduction of a memory size, simplification of a numerical approximation. This is the program of visualization, which is simple to use when displaying of results of modelling, creating animation and pictures for presentations and reports.

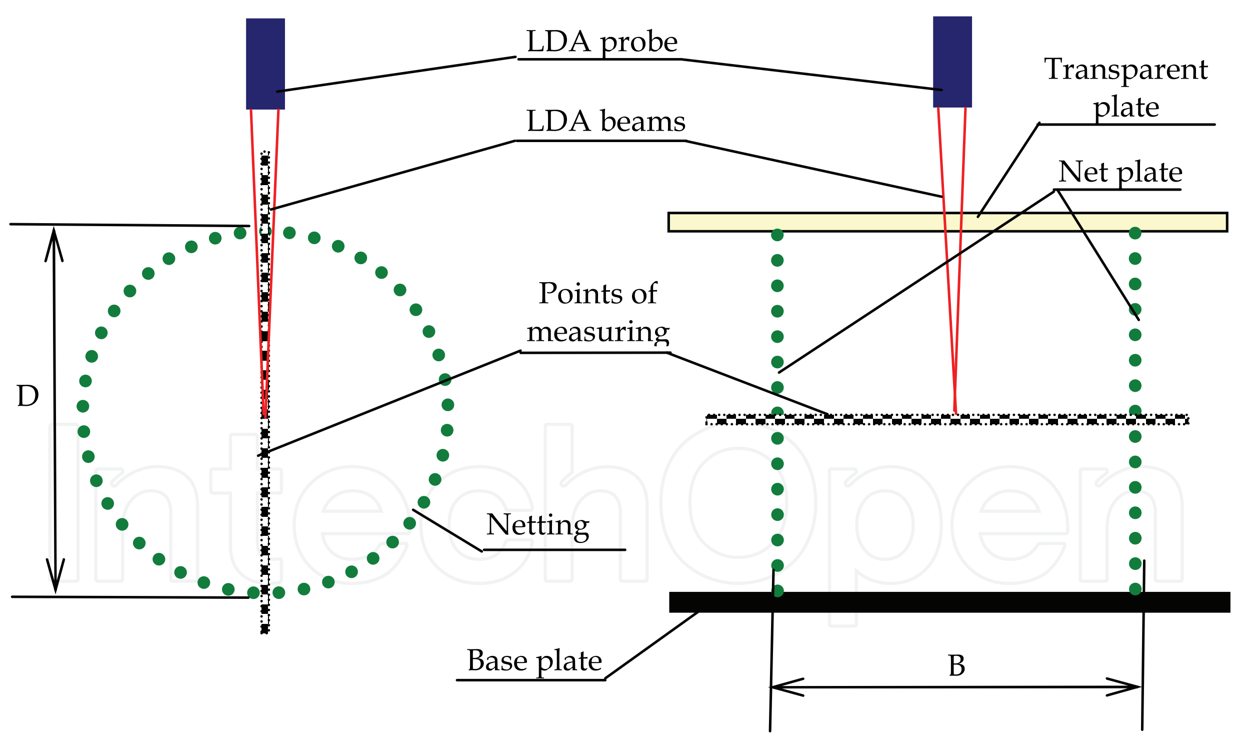
The software package FLUENT (http://www.fluent.com) is also the CFD program for a wide range of flows modelling: incompressible (low subsonic), compressed (transonic) and even super compressed (super-hypersonic). The program gives a set of the physical models allowing to predict parameters of laminar and turbulent liquid or gas flows with the big accuracy.

However modelling in these software packages is extremely labour-intensive process and takes a lot of time, which is necessary for definition of finite-element grid geometry and parameters according to the set model.

The program FEMLAB (http://www.comsol.com) is a package for solving of the equations modelling investigated system eliminates these problems. The FEMLAB software is the powerful interactive package for the modelling, enabling to solve all kinds of the scientific and technical problems based on Partial Differential Equations (PDE). The interface basing on the Java language has many advantages. Users can create model without programs writing. A library built in the program includes more than 200 completely solved and documentary firmware models, often meeting for solving waves distribution problems, for an analysis of internal combustion engines work, electromechanical systems, etc. After an automatic call of the built in model, users can change names of variables, to set factors and even to change a kind of the equations. The last feature is extremely difficult to realize in other packages. When the model chooses, for example, heating transfer, acoustics, hydrodynamics, diffusion, electromagnetism, the theory of elasticity or others, the program automatically sets the corresponding equations. As well, it is possible to enter the necessary equations independently. Moreover, users can combine and mutually connect any kinds of the phenomena, creating a multiphysical model.

The program FEMLAB is developed so that modelling of physical fields and connections between them can be carried out simply. There is an opportunity to solve the given system of PDE or to use specialized physical applied modes. These physical modes consist of the predetermined patterns and the interfaces of the user already installed with the equations and variables for specific areas of physics. Further, it is possible to create multiphysical model combining any number of these applied modes in the uniform applied description. It is possible in the FEMLAB system to expand easily usual models for one type of the physical phenomena in multiphysical models that solve the connected phenomena of physics, and make it simultaneously. Access to this opportunity does not demand a profound knowledge of mathematics or the numerical analysis that is very important for the engineer or the designer solving applied problems. Considering the described advantages of FEMLAB software and its latest version named as COMSOL Multiphysics the estimation of a net structure hydrodynamic field has been made.

4. Experiments on schematized net structures



a) cross section of a net cylinder b) cross section of SNS

Figure 5. Transformation from net structure to SNS

Researchers encountered problems in experiments to measure velocities everywhere inside a net structure because of a difficulty to penetrate with a log through meshes due to their small size especially in cod-end. Therefore, a method of flow velocities studies on schematized net structures (SNS) was proposed (Meyler, 1995). Taking into account that such constructions as the trawl cod-end has an axisymmetric shape, distributions of flow parameters (velocities and pressures) inside and around the structure have the axisymmetric character as well. Assuming these circumstances, flow velocities were studied by means of the Laser Doppler Anemometer (LDA) on a construction consistsing of two vertical net plates attached to two horizontal solid plates the upper of which is made of a transparent material. A view of “transformation” from a real net structure cross section to SNS cross section is shown in Fig. 5. Fig. 6 shows a scheme of a SNS test in the flume tank. It was assumed also that velocities distributions into limits of a very thin measured layer at the diametric section of a net cone or a net cylinder and into such layer at the central symmetric section of the SNS are identical. Net plates of the SNS was made of the same netting and the distance B between two plates was equal to the diameter D of the net cylinder or cone. Results of experiments on different SNS simulating net cylinders, cones and their combinations have shown (Meyler, 2000, 2005) a good convergence with measurements on real net structures.

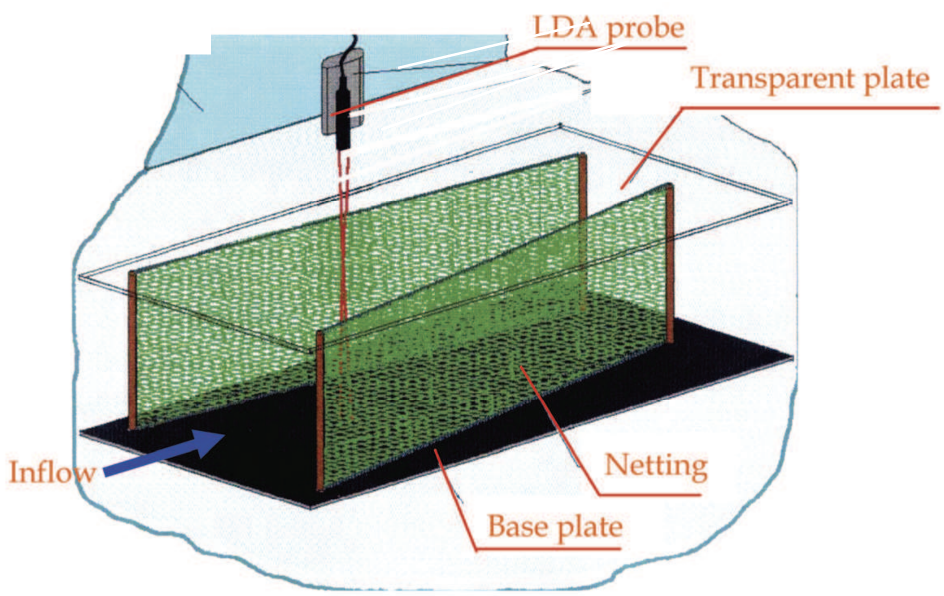


Figure 6. Scheme of the flume tank SNS “a cone” test

5. Preliminary estimation of FEMLAB (COMSOL Multiphysics) applicability

Naturally, considering the above described advantages of the given software package a question arises: whether it is possible to use it for counting a net structure hydrodynamic field? Preliminary evaluation of this method for a numerical estimation of a net structures hydrodynamic field was made (Meyler, 2006, 2007) on net cones and cylinders because they are the main elements of various real fishing gears. Taking into consideration assumptions accepted for SNS, cross sections of twines located at the thin layer were replaced with composition of circular cross sections. This composition simulates a longitudinal section of specified net structures: a cone, a cylinder and their combination. The flow around these sections is assumed to be equivalent to the flow around twines. Thus, the two-dimensional parallel-plane flow of an incompressible liquid around a system of the circular cylinder cross sections located perpendicularly to the flow direction was considered. It is known, that the incompressible liquid flow is described by Navier-Stokes equations consisting of the momentum (balance) equation at performance of the mass preservation law and an incompressibility condition:

$$\begin{cases} \rho\dfrac{\partial \mathbf{u}}{\partial t} - \rho(\mathbf{u}\cdot\nabla)\mathbf{u} = -\mathrm{grad}\ p + \mathrm{div}(\eta\cdot(\nabla\otimes\mathbf{u} + \mathrm{grad}\ \mathbf{u})) + \mathbf{F} \\ \mathrm{div}\ \mathbf{u} = 0 \end{cases} \tag{1}$$

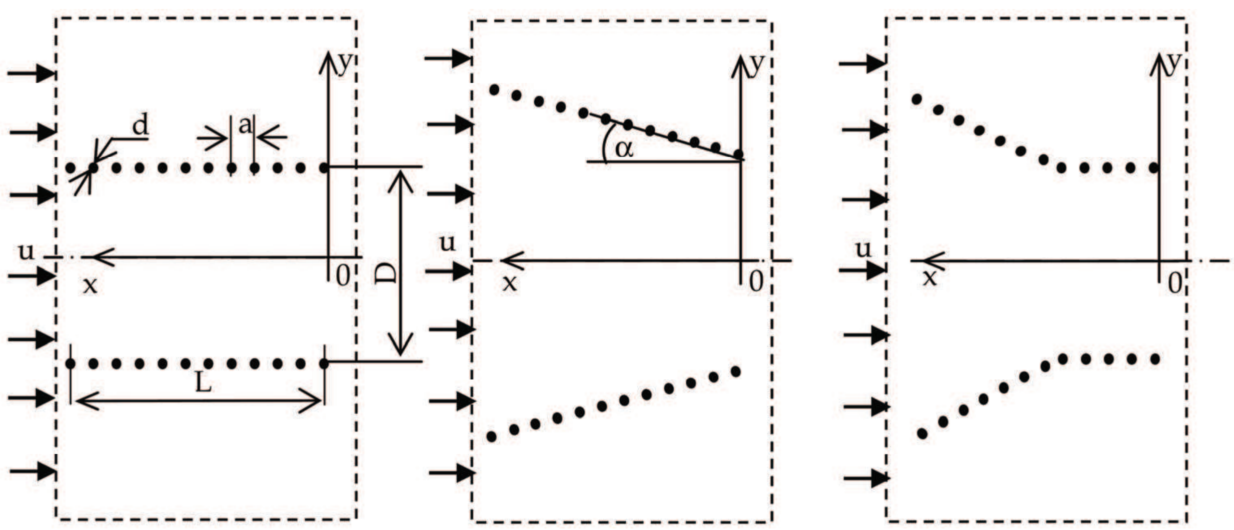
$$\mathbf{u} = \mathbf{i}_x\cdot u + \mathbf{i}_y\cdot v;\quad \nabla = \mathbf{i}_x\frac{\partial}{\partial x} + \mathbf{i}_y\frac{\partial}{\partial y};$$

where: ρ - liquid density (ML^{-3}); $\mathbf{u}$ - vectorial velocities field (LT^{-1}); $\mathbf{i}_x, \mathbf{i}_y$ - axes unit vectors; p – pressure ($ML^{-1}T^{-2}$); η - dynamic viscosity ($ML^{-1}T^{-1}$); $\mathbf{F}$ – field of vectorial force ($ML^{-2}T^{-2}$); ∇ - vectorial differential operator.

The COMSOL Multiphysics software uses these equations for the decision of flow dynamics problems. The schemes of each "net" constructions model are shown in Fig. 7. These schemes appear in the Graphic User Interface (GUI) of the COMSOL Multiphysics. The models had the same characteristics as those in experiments. Cross sections of the netting twine were presented in GUI as so called "solid" round bodies placed in that order to get longitudinal sections of a "net" cylinder, a "net" cone, and their combinations. It is possible to change a model "permeability" varying with the distance between "solid" bodies.

Characteristics of models were the following:

- Diameter of a circular cross section was equal to a diameter of a real twine $d = 2.5\cdot10^{-3}$ m;
- Distances between these sections were equal to a mesh size of a real netting $a = 25\cdot10^{-2}$ m;
- Length of models L=1.4m;
- Inflow model diameter D = 0.3m.



a) cylinder b) cone α ≈10° c) cylinder + cone α ≈ 10°

Figure 7. Scheme of the net models

To calculate a hydrodynamic field in/around these objects basing on Navier-Stokes's equations the field of the flow velocity, i.e. the "N-S" subdomain was created in GUI. Fig. 7 shows this subdomain as a dotted rectangular. The left margin of the subdomain is considered as the flow inlet and the right margin as the flow outlet. Values of the water density ρ and the water dynamic viscosity η as well initial inlet longitudinal u_0 and transversal v_0 flow velocities and initial hydrostatic pressure p_0 are pre-assigned for this subdomain.
The subdomain boundary settings i.e. boundary conditions at the "N-S" subdomain were the following:

- an inlet margin – initial velocities of a water flow, expressed as "inflow/outflow", for the model were equal to real inlet velocities $u=u_0=1.0$ m/s, $v_0=0$;
- an outlet margin – an outgoing flow expressed as "outflow/pressure";
- at top/down margins – a free flow expressed as "neutral";
- at each "solid" body - a non-leaking body contour expressed as "no slip".

As the analysis has shown, these borders are necessary to place on significant distances from the model, so that "flow" borders do not influence a field of velocities around the model. As it was mentioned, a big advantage of the software is the automatic mode of a finite-element grids generation under the set boundary conditions of the flow. Fig. 8 shows the fragment of a grid near to a "twine". It is possible to see that boundary conditions "no slip" form the tighter grid.

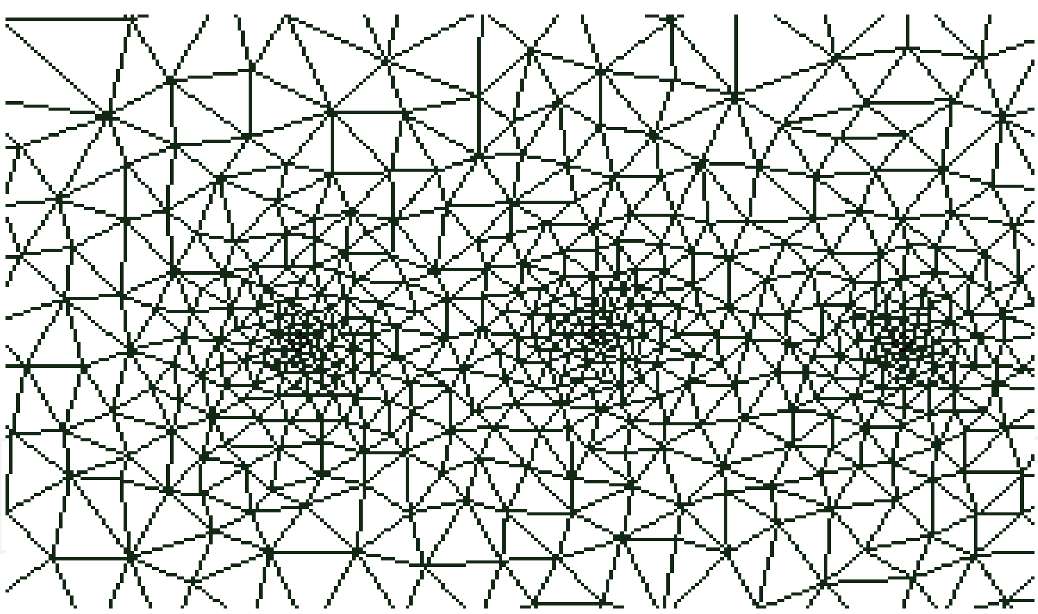


Figure 8. Fragment of a finite-element grid

Fig. 9 shows the calculated velocities distribution for the model – "a close net cylinder", i.e. there is a "net wall" at the outlet. Fig. 10 demonstrates comparison between calculated dependences of the relative longitudinal velocity (solid lines) and experimental data (symbols of the same color), received in research of SNS from the relative longitudinal x/L and transversal $\overline{y} = 2 \cdot y/D$ coordinates. Comparison of calculated and experimental data testifies to their insignificant divergence.

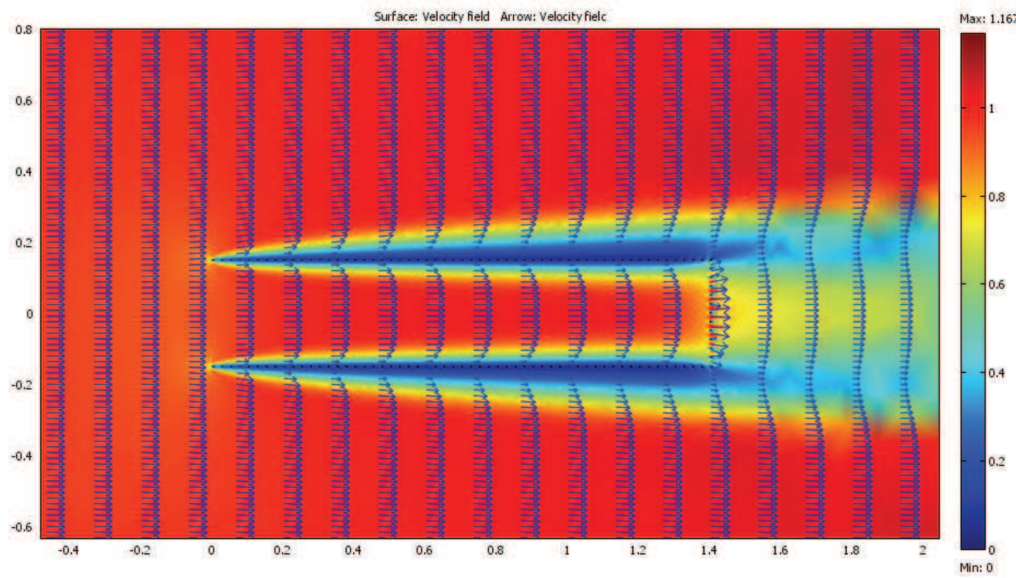


Figure 9. Velocity distributions around a “close net cylinder”

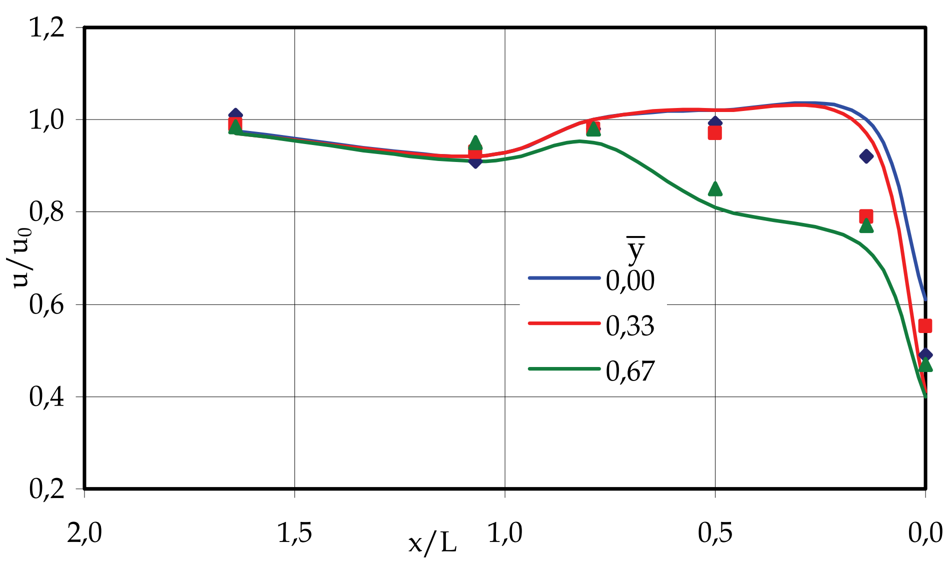
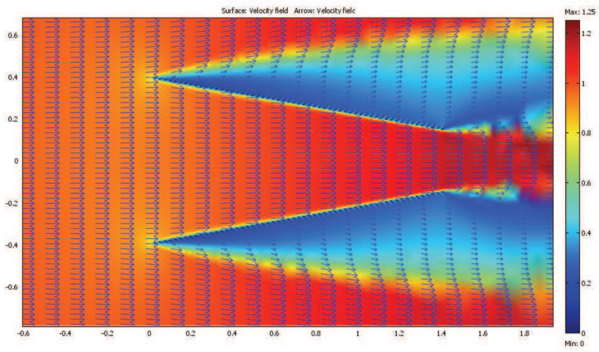
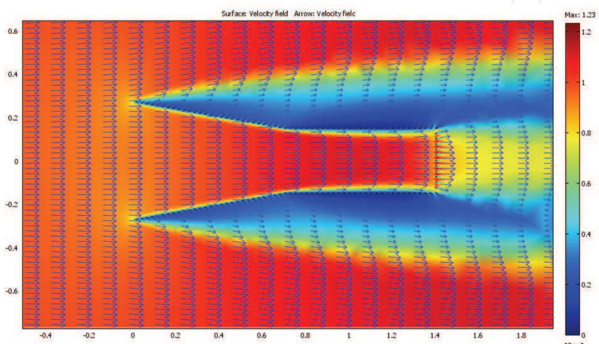


Figure 10. Calculated and measured data comparison for a “close cylinder”

Experimental and calculated data have satisfactory convergence for other variants of models. Velocity distributions for some of them are shown on Fig. 11. These figures give a possibility to estimate both values of flow velocities and features of the flow. It is possible to assume that calculating and summarizing parameters of hydrodynamic fields for similar "net" elements allow estimating such field for a trawl in total. Results of the calculated and experimental data comparison allow to make a conclusion about an opportunity to use the COMSOL Multiphysics software for estimation of net structures hydrodynamic fields.



a) “an open net cone” α=10°



b) “a net cone + a close net cylinder”

Figure 11. Flow velocities distribution for “net” models

Fig. 12 illustrates the flow into rear net cylindrical part of the cod-end with “a catch” simulation. The fish simulation was created with “solid” bodies given as cylindrical cross sections. It is seen that “the catch” gives a backwater flow and, so called, “locking” effect. It permits to have a possibility for a fish escape from the trawl or make “meshing” of the cod-end that is to block meshes of the net.

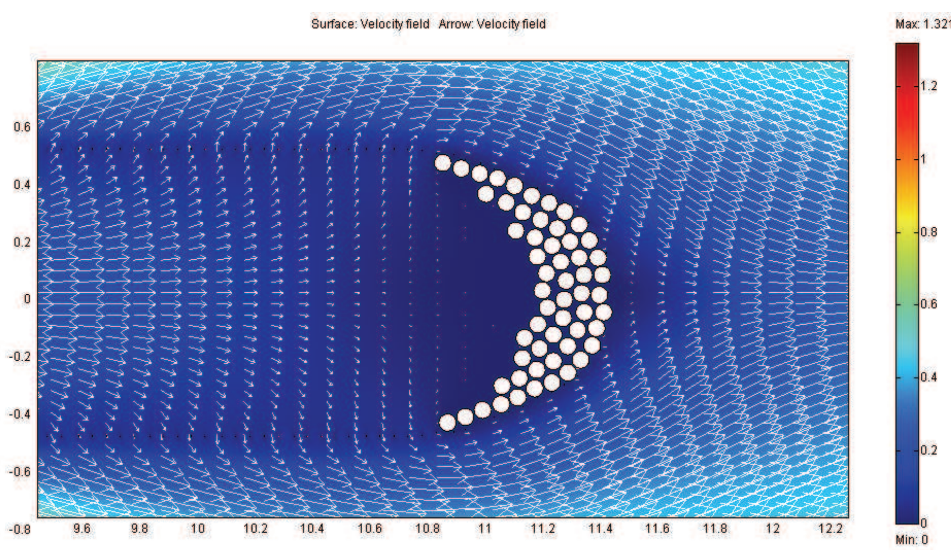


Figure 12. Speed distribution with a “catch” imitation in a cod-end

6. A permeable surface concept

However, such kind of simulations faces some difficulties. First, it is complex big sizes "net" models drawing in GUI because of many "solid" bodies, especially when permeability of netting is necessary to change. Secondly, too long time of calculations. Sometimes a PC operative memory had not enough capacity. Therefore, this approach can be sufficient for simple models simulating. In this connection works conducted for description of an ideal or viscous liquid flows around of permeable or perforated bodies, surfaces, envelopes and parachute's canopies are of interest. Use of these theoretical researches results for fishing net structures depends on effects which are on a permeable surface, in particular on permeability coefficient values which are obvious more for netting, than for a parachute canopies fabric. In addition, the account of viscosity and formation of a vortical structure behind a permeable surface is obviously important. These theoretical researches are based on the known physical laws described by the known mathematical equations and correlations that give the possibility to use them for fishing gears application. The problem of a flow through a permeable surface has been solved as an application to the theory of a parachute (Rakhmatullin, 1950). It was accepted an assumption about a uniform distribution of velocities and pressures at a small area of the permeable surface and that a normal component of a flow velocity and a pressure difference are connected with square-law function. A tangential component of a flow velocity can have a break. It means that on the uniform permeable surface velocities and pressures can have breaks. Because the uniform permeable surface is not a liquid surface, a liquid particle receives some impact flowing through this surface. Fig. 13 shows the scheme of the flow through a permeable surface.

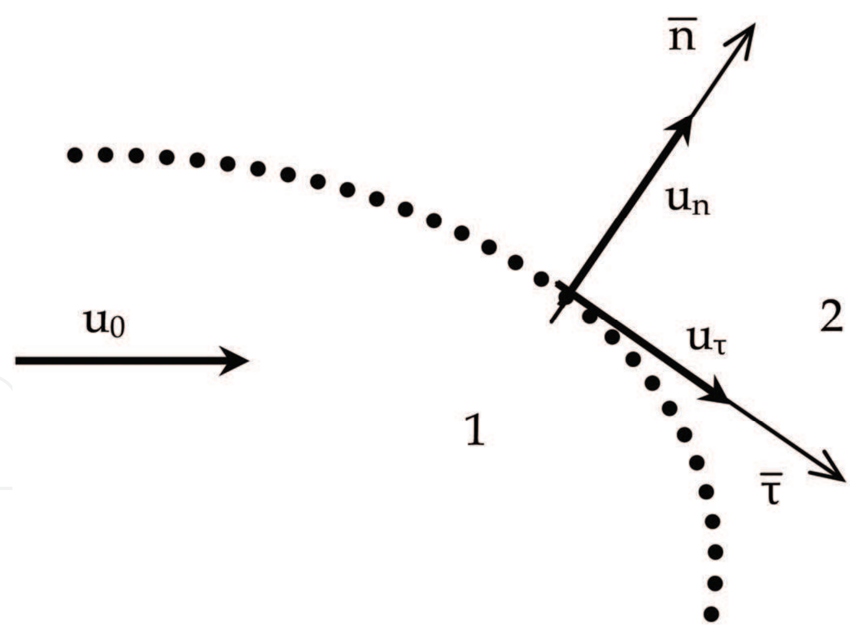


Figure 13. Scheme of a flow through the permeable surface

A flow through a permeable surface is defined with laws:
Mass conservation:

$$\rho_1 \cdot u_{1n} = \rho_2 \cdot u_{2n} \tag{2}$$

Impulse conservation:

$$\begin{aligned}\rho_1 \cdot u_{1n} \cdot (u_{2n} - u_{1n}) = p_1 - p_2 - R_n, \\ \rho_1 \cdot u_{1n} \cdot (u_{2\tau} - u_{1\tau}) = -R_\tau,\end{aligned} \tag{3}$$

Energy conservation:

$$\rho_1 \cdot u_{1n} \cdot \left(\frac{u_2^2}{2} - \frac{u_1^2}{2} \right) = p_1 \cdot u_{1n} - p_2 \cdot u_{2n} - \overline{R} \cdot \frac{\overline{u}_1 + \overline{u}_2}{2}, \tag{4}$$

where: ρ_1, ρ_2 – water densities; u_{1n}, u_{2n} – velocity projections on the normal; u_1, u_2 - velocity modules; p_1, p_2 - pressures; R_n, R_τ - normal and tangential components of a flow force acting on a permeable surface unit; $\overline{R}$ - the resulting vector of R_n, R_τ.

Areas "1" and "2" in Fig. 13 and indexes in equations (2 – 4) define flow parameters in front of the permeable surface and behind it. This model needs in evaluation of the permeable surface different details (twines and knots of the net) impact on the water flow. It presents difficulties because of the multiple diffraction and interference of a flow disturbance around twines. Interaction between an incident flow and the permeable surface depends on meshes distribution, their number and inclination to the velocity vector. It is possible to take assumption that the flow around a net surface has some features:

- a shape conservation, i.e. the net structure has factually unchangeable shape under the flow impact;
- a net structure shape creation, i.e. the structure «chooses» itself such shape under the flow that a pressure difference on its surface has a constant value;
- a net structure has a definite permeability that is a main factor of a flow stability around the net structure.

Boundary conditions are defined taking into consideration real features of the water flow through the net surface:

for the fixed permeable surface (for example, a set long line or a seine):

$$\text{on the surface } \frac{\partial \varphi}{\partial n} = u_i; \quad \text{in infinity } \frac{\partial \varphi}{\partial x} = u_0, \quad \frac{\partial \varphi}{\partial y} = \frac{\partial \varphi}{\partial z} = 0.$$

for a moving permeable surface (for example, a trawl or a cod-end):

$$\text{on the surface } \frac{\partial \varphi}{\partial x} = -u_0 \cdot \cos(u \wedge n) + u_i; \quad \text{in infinity } \frac{\partial \varphi}{\partial x} = \frac{\partial \varphi}{\partial y} = \frac{\partial \varphi}{\partial z} = 0.$$

where: φ – a velocities potential.

7. Use COMSOL Multiphysics for hydrodynamic fields around permeable net structure models calculation

In order to have a possibility to vary easier netting permeability for calculations of hydrodynamic fields of various "net" structures the COMSOL Multiphysics software functions give an opportunity to combine solving of different physical fields. In particular, parameters of flows through permeable surfaces or bodies can be defined on the basis of Brinkman's equations. These equations describe flow in porous (permeable) media where momentum transport by shear stresses in the fluid is of importance and can be applied when modeling combinations of porous media and free flow. The flow field is determined by the solution of the momentum balance equations in combination with the continuity equation:

$$\begin{aligned} &\rho\frac{\partial \mathbf{u}}{\partial t}-\nabla\cdot\eta\left[\nabla\mathbf{u}+(\nabla\mathbf{u})^{T}\right]-\left(\frac{\eta}{k}\mathbf{u}+\nabla p-\mathbf{F}\right)=0 \\ &\nabla\cdot\mathbf{u}=0 \end{aligned} \tag{5}$$

where: k – permeability of the porous structure (L^2), other denotations are the same as in (1).
The Brinkman equations account for momentum transport through viscous effects and through pressure gradients in porous media. It was suggested that the porous media is a permeable surface of netting for a fishing gear. Therefore, it is necessary to define a value of the permeability for simulated net structure model using this approach. Naturally, as the first stage of this approach using the model of the "close net cylinder" having the same dimensions as for the corresponding SNS was studied.
Fig. 14a shows this "net" structure model drawn in GUI with real dimensions. The thickness of the contour is equal to the SNS twine diameter. The "N-S" subdomain has a grey color and the "Br" subdomain is in red one. Thus, in this case, two subdomains are necessary to be used for flow velocities and pressures calculation. The "N-S" subdomain describes a flow field in/around the model basing on Navier-Stokes's equations. The contour is the "Br" subdomain that describes flow through the permeable netting contour basing on Brinkman's equations. For these subdomains values of water density ρ (kg/m^3) and water dynamic viscosity η (kg/m·s) are given. The contour permeability k_{mod} has to be given for the "Br" subdomain. The real value of net structure permeability:

$$k_{real} = F_{real} - F_{tw}\,, \tag{6}$$

where F_{real} - the net structure common lateral area; F_{tw} - the common twine shaded area.
The ratio F_{tw}/F_{real} is assumed to be the same both for the real net structure and for the model. Taking into account that the common model contour area $F_{mod} = L\cdot t$, the value of the model contour permeability:

$$k_{mod} = L\cdot t\cdot(1-F_{tw}/F_{real})\,, \tag{7}$$

where L - the common model contour length; t - the model contour thickness.
To calculate flow velocities in/around the "net" model u_{ns}, v_{ns}, boundary conditions at the "N-S" subdomain are the following:

- an inlet margin – "inflow/outflow",
- an outlet margin – "inflow/outflow",
- top/down margins – "neutral";

Initial flow velocities at the "N-S" subdomain inlet margin: $u = u_0 ; v = v_0$. Flow velocities at the "Br" subdomain inlet/outlet margins, i.e. at the contour's inside/outside borders, are $u = u_{ns}$ and $u = u_{br}$ accordingly.

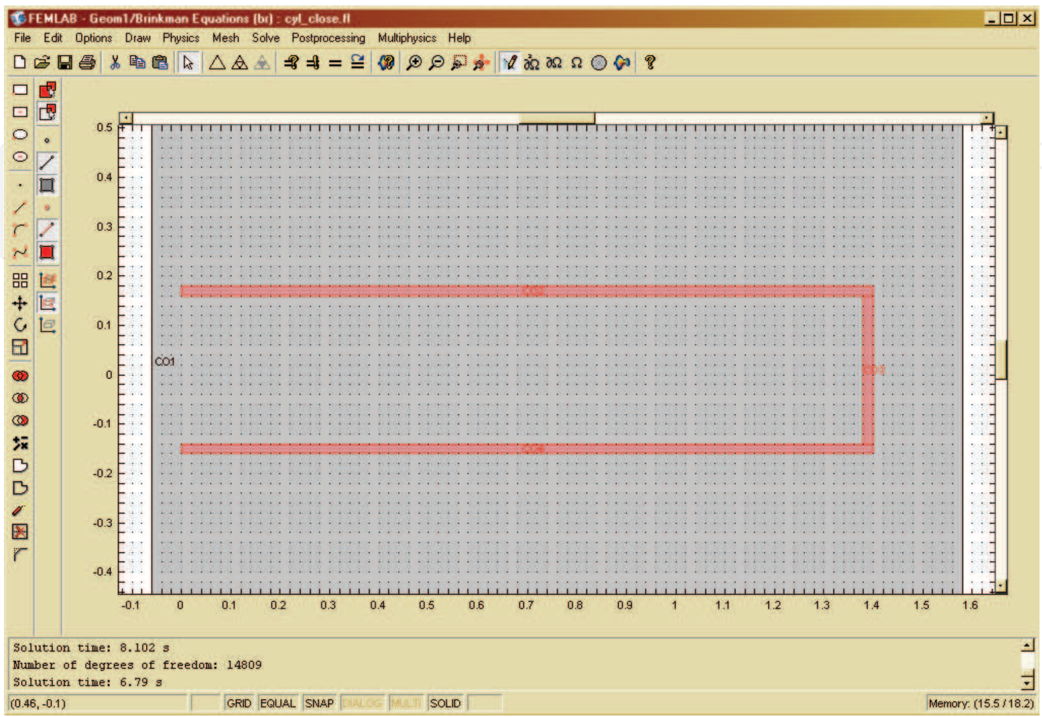


Figure 14. Net permeable model "a close cylinder" in GUI

Fig. 15 illustrates the flow distribution in/around this model. A comparison with the flow distribution in Fig. 9 shows rather similar views. Some difference is observed in the velocity dependence character.

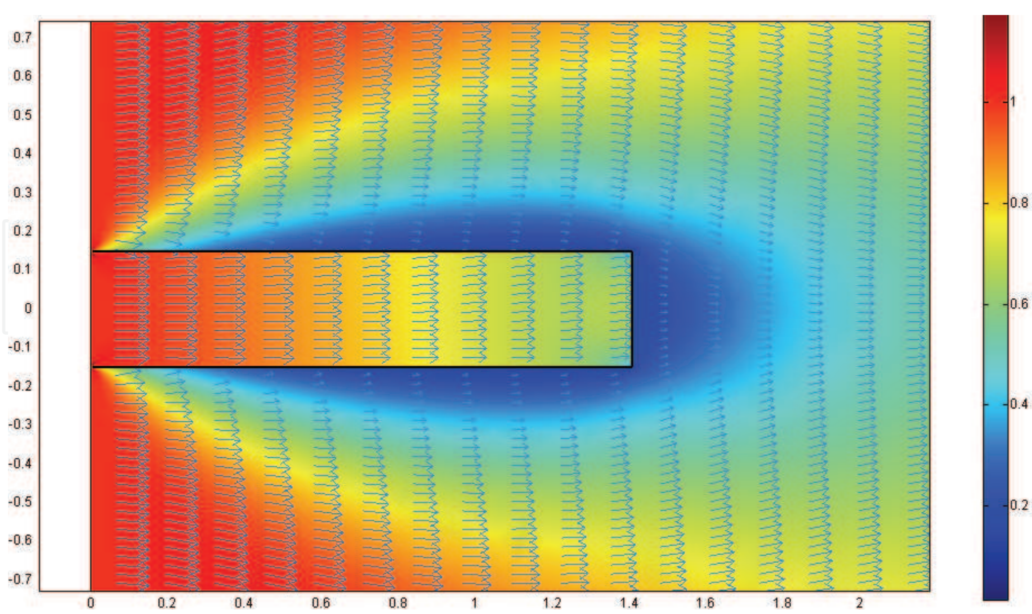


Figure 15. Flow velocities distribution for "a close cylinder" model

Obviously, the correct definition of the permeability value corresponding to real netting conditions can give a possibility to reduce so called "scale effect" and to simulate net structures. It is important that this approach allow changing values of permeability along the model if it is created in GUI as a multicomponent assembly. Fig. 16 shows the velocities field for the model "open net cone α=10°". It is possible to compare it to Fig. 11a.

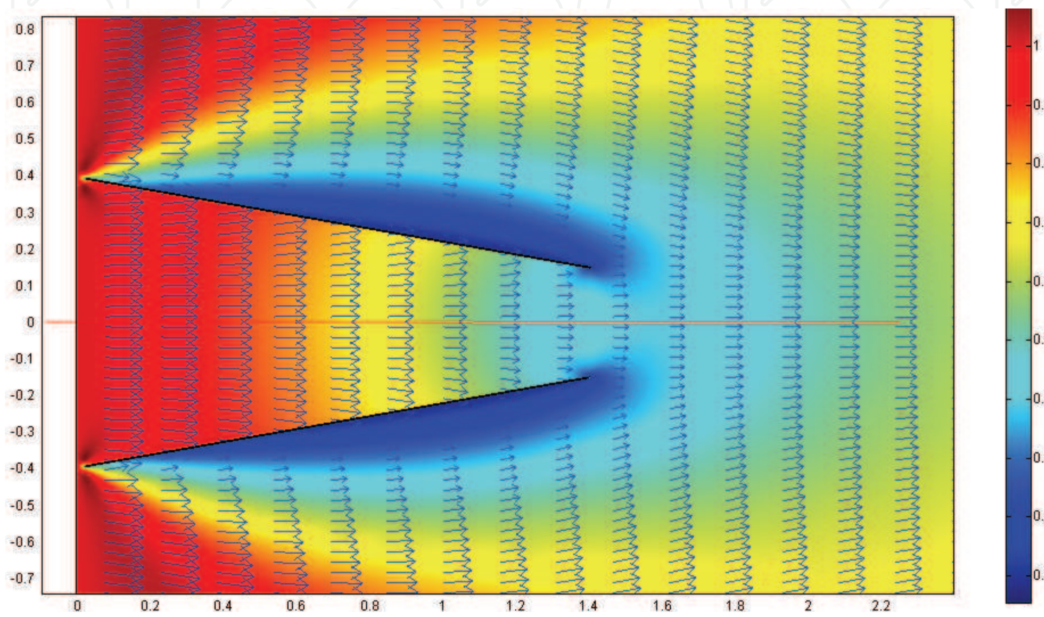


Figure 16. Flow velocities distribution for "an open net cone α=10°" model

To calculate a pressure field in/around the "net" model, which are p_{ns} values boundary conditions at the "N-S" subdomain are the following:

- inlet margin – "normal flow/pressure",
- outlet margin – "outflow/pressure",
- top/down margins – "neutral";

An initial pressure at the "N-S" subdomain inlet margin: $p=p_0$. The value of p_0 is equal to the atmospheric and hydrostatic pressures sum for the real net structure. Pressures at the "Br" subdomain inlet/outlet margins, i.e. at the contour's inside/outside borders, are $p=p_{ns}$ and $p=p_{br}$ accordingly.

As an example of real net structure model the contour of a cod-end model tested in the flume tank is shown in Fig. 17. When the model was tested its inlet margin, so called "mouth", was equipped with a rigid ring in order to keep a shape under flow.

Using all mentioned parameters of "N-S" and "Br" subdomains and calculating a value of permeability for the cod-end model the flow velocities distribution was defined and presented in Fig. 18. It is possible to see that colours change from red to deep blue, i.e. value of velocities decreases to the rear part of the model. Fig. 19. shows effect filling the cod-end with a "catch". The "catch" simulates by the "solid body" placed into the contour and having a shape similar to real catch. The distribution of the velocities is changed. The velocity decrease is stronger in the rear part of the model and just in front of the "catch" velocities are equal to zero.

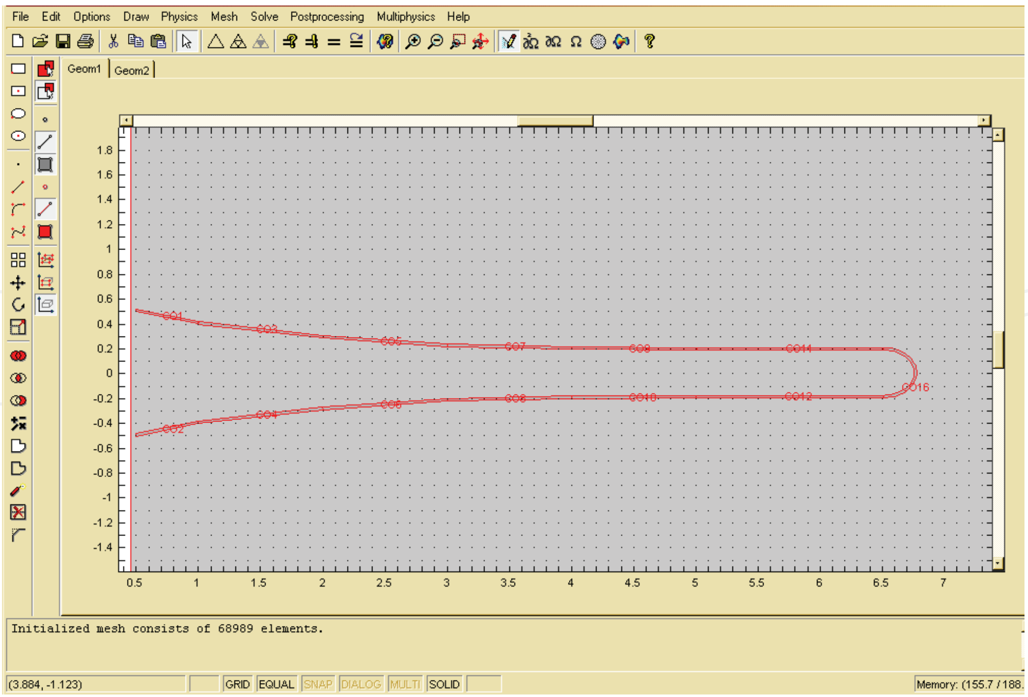


Figure 17. Net permeable model "a cod-end" in GUI

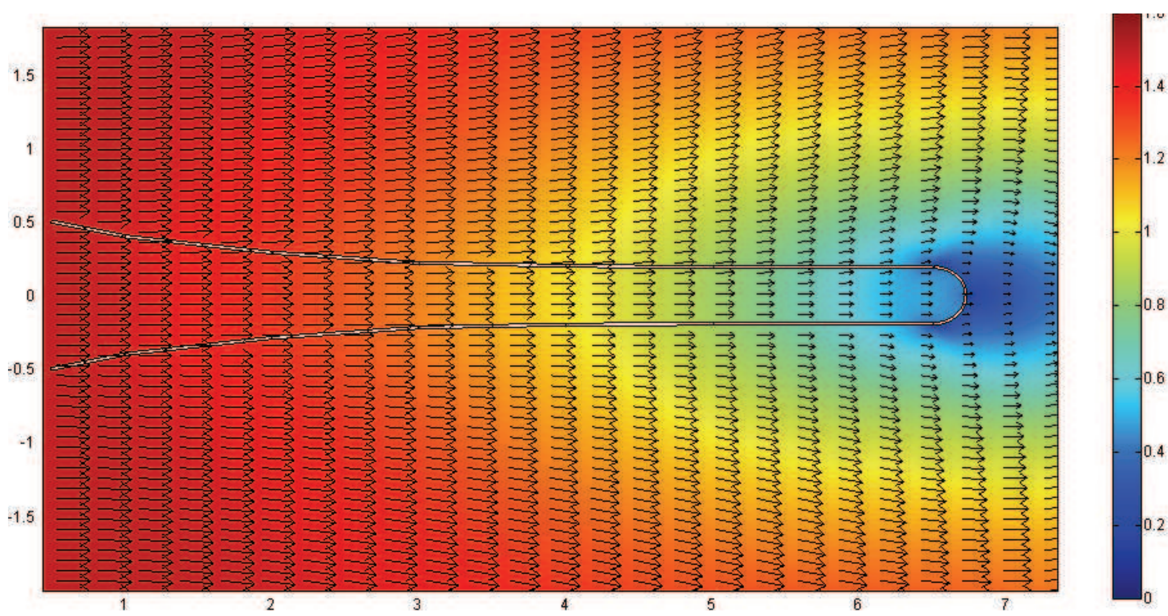


Figure 18. Flow velocities distribution for "a cod-end" model

As it was mentioned in the survey, in reality the cod-end with caught fish has another shape due to its mechanical characteristics. An approximate shape and velocities distribution is shown in Fig. 20. Here it was an artificial drawing of the cod-end shape, but it is necessary to find a solution to predict fishing gears shape under the flow and with a catch taking into account mechanical features of netting.

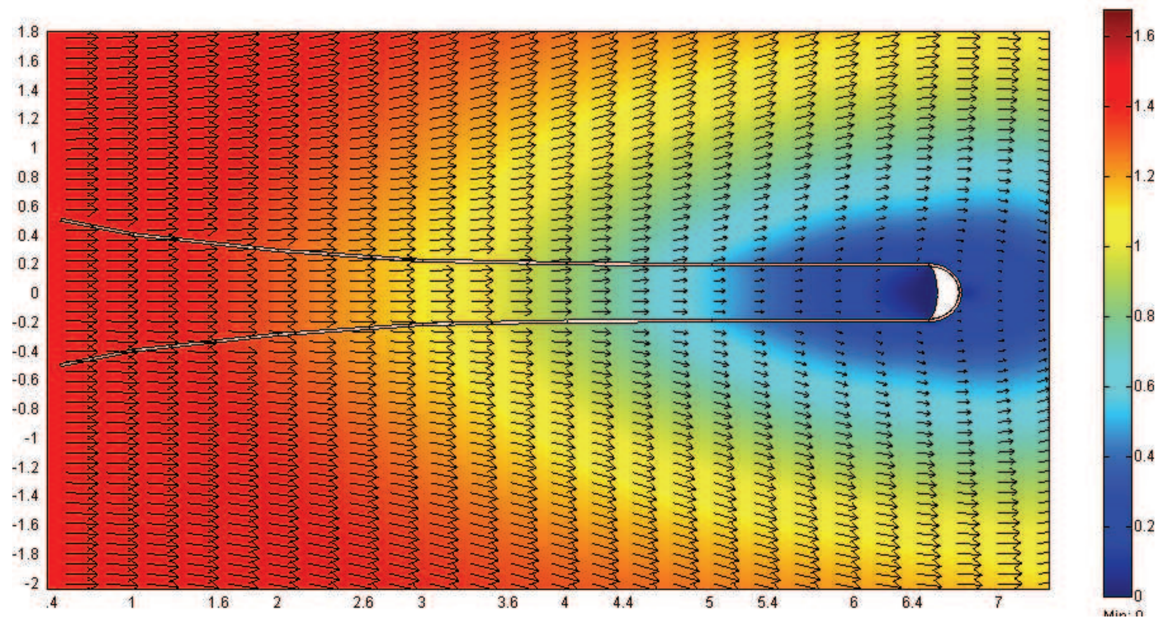


Figure 19. Flow velocities distribution for “a cod-end with “catch” model

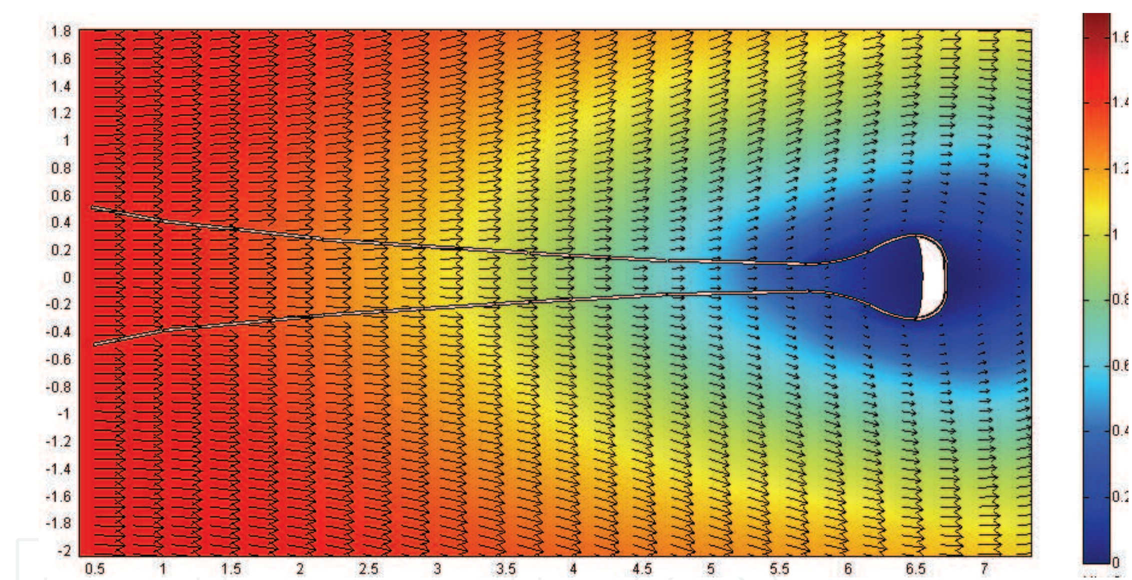


Figure 20. Flow velocities distribution for “a cod-end with “catch” model

8. Conclusion

The presented work considers two approaches to use a CFD method for modelling the hydrodynamic field of fishing gear net structures. These approaches are based on the COMSOL Multiphysics software use allowing to model physical fields of various bodies flow of a flow. For an estimation of applicability of the software results of experimental researches on schematized net structures (SNS) carried out in the flume tank have been used.

The SNS contour is represented as combinations of circular cross-sections in the first modelling approach. Such contour is used as the calculated object simulating twines cross-sections. Calculation of flow velocities around "solid" bodies located so that to present a net structure section made basing on Navier-Stokes equations. Comparison of a calculated dependence of longitudinal flow velocities to the measured velocities values has shown rather satisfactory convergence of data. However, the application of the given approach faces long time for solving when the contour consists of a significant number of the simulating bodies.

The second approach uses a multiphysical statement of a problem in the software. A "net" contour represents a "permeable surface" simulation of the net structure model. A flow through the permeable surface is described by the Brinkman equations. Two subdomains are set for calculation of the structure of multiphysical hydrodynamic field:

1. The Navier-Stokes's subdomain,
2. The Brinkman's subdomain.

It is necessary to set a liquid density, its dynamic viscosity and an initial pressure for the first subdomain. For the second one it is necessary to set the permeability of the contour surface as well. Solution time using the second approach is much less than in the first one. For example, "a closed cylinder" model calculation took about 1 hour in the first case and several minutes in the second case. It is possible to see in Figs 9 and 15 the velocities field picture differs for these two approaches. A reason can be connected with correct given value of net structure permeability that depends on the flow influence on the net structure.

It is necessary to notice, that the second approach enables changes of a net contour permeability if it consists of several elements. This circumstance can be used, for example, in case of imitation of net surfaces blocking up with a catch or for calculation net structures consisting of net panels having various mesh sizes, for example for a fishing trawl.

The systematized comparison of calculated and experimental data for more reliable estimation of applicability of both methods is necessary.

In addition, researches of applicability of the method are necessary to carry out with the purpose of revealing of influence of a hydrodynamic field on the net structure shape.

9. References

Baev, L. & Belov, V. (1987). Determination of a trawl cod-end model velocity field. *Rybnoe khozyaistvo*, № 1, 1987. pp. 47–48 (in Russian);

Belov, V. (1987). Water flow in a trawl. *In proceedings of the all-Union scientific-technical workshop on hydrodynamics and fishing gears designing*, Kaliningrad, 1987, pp. 110-115 (in Russian);

Belov, V. (2002). Water flows in trawls. *Rybnoe khozyaistvo*, № 1, 2002, pp. 51–52 (in Russian);

Gabriuk, V. & Chernetsov, V. (1985). A raft-catamaran for hydromechanics of a trawl and its rigging investigations. *Rybnoe khozyaistvo*, № 5, 1985, pp. 65–66 (in Russian);

Enerhaug, B.; Gjøsund, S.H. & Hansen, K. (2001). Experimental, numerical and analytical studies of flow through reticulate and solid cones, *Proceedings of the 5-th international workshop "Methods for the development and evaluation of maritime technologies (DEMaT '01)"*, pp. 43–59, Rostock, November, 2001;

Enerhaug, B. (2005). Flow through fine-meshed pelagic trawl. *Proceedings of the 7-th international workshop "Methods for the development and evaluation of maritime technologies (DEMaT '05)"*, pp. 153–163, Busan, November, 2005;

Fredheim, A. & Faltinsen, O.M. (2001). A numerical model for the fluid structure interaction of a three-dimensional net structure, *Proceedings of the 5-th international workshop "Methods for the development and evaluation of maritime technologies (DEMaT '01)"*, pp. 61–82, Rostock, November, 2001;

Fuwa, S., Fujita, S., Kumasawa, T. & Hirayama, M. (2007). Flow distribution in a brailer codend, *Proceedings of the 8-th international workshop "Methods for the development and evaluation of maritime technologies (DEMaT'07)"*, pp. 11–21, Rostock, October, 2007;

Higo, N. (1964). Studies on the drag net - I; An increase of the current velocity inside the net, *Memoirs of the faculty of fisheries, Kagoshima University*, vol. 13, 1964, pp. 78-92;

Higo N., Tokunaga Y. & Fuwa S. (1974). Studies on the drag net - III; Current velocity reduction at the inside of the net, *Memoirs of the faculty of fisheries, Kagoshima University*, vol. 23, 1974, pp. 29-34;

Imai T. (1974). Distribution of flow-speed inside the net and performance of the net-shape, *Memoirs of the faculty of fisheries, Kagoshima University*, vol. 23, 1974, pp.45-55;

Korotkov, V. & Meyler, L. (2000). An improved trawl cod-end design. *Rybnoe khozyaistvo*, № 4, 2000, p. 46, (in Russian);

Korotkov, V. & Meyler L. (2001). Hydrodynamic investigations, designing and marine tests of a trawl codend with the improved conditions of a catch concentration, *Proceedings of the 5-th international workshop "Methods for the development and evaluation of maritime technologies (DEMaT '01)"*, pp. 175–187, Rostock, November, 2001;

Kostiukov, V. & Shevchenko, A. (1983). A study of a trawls net shells hydrodynamic field. *Rybnoe khozyaistvo*, № 9, 1983, pp. 63-64 (in Russian);

Kroeger, M. (1984). Some results of flow measurements on a full-scale pelagic trawl, *ICES-CM-1984/B: 27, Council Meeting Collected papers*. pp. 11-19, Copenhagen, 1984;

Meyler, L. (1995). Flow speed investigation on schematized net constructions, *Report № 94/71, IFREMER*, France, 1995, 95 p.

Meyler, L. (2000). A velocity field in a schematized net structure, *Proceedings of the international scientific-technical conference "The 70-th anniversary of the Kaliningrad State Technical University"*, Vol.2, pp. 346-347, Kaliningrad, 2000, (in Russian);

Meyler, L. (2005). Schematized net structures for fishing gears velocity fields studying. *Rybnoe khozyaistvo*, № 4, 2005, pp. 63–64, (in Russian);

Meyler, L. (2006). Estimation of flow speed and pressure distribution in a fishing trawl cod-end, *Comsol Users Conference*, *User Presentations and Proceedings CD*, Prague, 2006;

Meyler, L. (2007). Use of COMSOL Multiphysics for a hydrodynamic field of net fishing gears calculation. *Rybnoe khozyaistvo*, № 2, 2007, pp. 105–107, (in Russian);

Miziurkin, M. & Kostiukov, V. (1982). About hydrodynamics of a trawl. *Rybnoe khozyaistvo*, № 6, 1982, pp. 66–67, (in Russian);

Moderhak, W. (1993). Some problems of water flow through trawl codend. *ICES Council Meeting, Fish Capture Committee, (Collected papers) L, ICES-CM-1993/B:11*, Copenhagen, 1993, 6 p.

O'Neill, F.G. (2003). A theoretical study of the factors which influence the measurement of fishing netting mesh size. *Ocean Engineering*, № 30, 2003, pp. 2053-2063;

Paschen, M. & Winkel, H-J. (1999). Flow investigations of net cones, *Proceedings of the 4-th international workshop "Methods for the development and evaluation of maritime technologies (DEMaT '99)"*, pp. 197–216, Rostock, November, 1999;

Paschen, M., Köpnick, W. & Winkel, H-J. (2001). Flow investigations of net cones, *Proceedings of the 5-th international workshop "Methods for the development and evaluation of maritime technologies (DEMaT '01)"*, pp. 189–205, Rostock, November, 2001;

Priour, D. & Herrmann, B. (2005). Catch shape in cod-end, *Proceedings of the 7-th international workshop "Methods for the development and evaluation of maritime technologies (DEMaT '05)"*, pp. 41–57, Busan, November, 2005;

Rakhmatullin, X. (1950). Flow around a permeable body. *Bulletin of the Moscow State University. Series of physic-mathematical and natural sciences,* Moscow, № 3, 1950, pp. 41 – 55 (in Russian);

Rehme & Scheel, (1988). Experimental investigations to flow through trawl net cones. *Seewirtschaft*, Berlin, Vol. 20, № 1, 1988, pp. 39-42 (in German);

Scharping, K.-D. Contribution for flow investigations in a fishing net, *Fischerei-Forschung*, № 12, 1974, pp. 57-63 (in German);

Stengel, H. & Fischer H.-J. (1964). Results of trawl model flow studies in a wind tunnel. *Fischerei-Forschung*, № 2, 1964, pp. 57-100 (in German);

Thiele, W. (1997). Flow measurements in a trawl, *Proceedings of the 3-rd international workshop "Methods for the development and evaluation of maritime technologies (DEMaT '97)"*, pp. 19–31, Rostock, November, 1997;

Patursson, Ø., Swift, M. R., Tsukrov, I., Baldwin, K. & Simonsen, K. (2007). Modeling Flow Through and Around a Net Panel Using Computational Fluid Dynamics, *OCEANS'06 MTS/IEEE - Boston*, Boston, Massachusetts, USA, September 2006;

Vincent, B. & Marichal, D. (1996). Computation of the flow field in the cod-end, *ICES FTFB Working Group Meeting*, 1518, April, Woods Hole;

Winkel, H.-J. & Paschen, M. (2001). Wind tunnel tests for fishing gear development – methods and limits, *Proceedings of the 5-th international workshop "Methods for the development and evaluation of maritime technologies (DEMaT '01)"*, pp. 29–41, Rostock, November, 2001;

Ziembo, Z. (1974). Distribution of speed of water flowing through a trawl, *Prace Morskiego Institutu Rybackego*, Vol. 17, Ser. B, 1974, pp. 7-17;

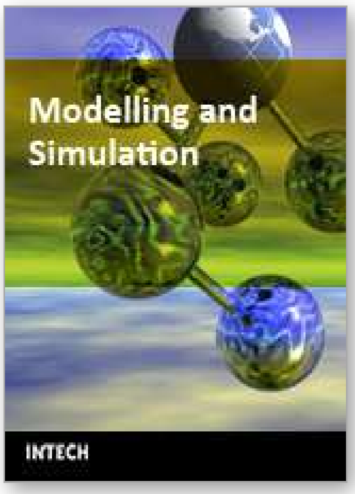
Ziembo, Z. (1987, a). Correlations among velocities of water flow through the trawl, *Biuletyn Morskiego Institutu Rybackego*, Vol. 18, № 5-6, 1987, pp. 14-20;

Ziembo, Z. (1987, b). Flow of water in trawls, *Scottish Fisheries Research*, SFRT, 66, 1987, Aberdeen;

Ziembo, Z. (1988). Mean velocity of bound vortex flowing around mesh bars of trawl nets, *Biuletyn Morskiego Institutu Rybackego*, Vol. 19, № 1-2, 1988, pp. 62-71;

Ziembo, Z. (1989). Factors affecting longitudinal velocity of water flow inside the trawl, *Biuletyn Morskiego Institutu Rybackego*, Vol. 20, № 3-4, 1989, pp. 55-59;

Ziembo, Z. (1993). Turbulent mixing of water flow within trawl walls and method of calculating component force of reaction, parallel to direction of motion, *Biuletyn Morskiego Institutu Rybackego*, № 130, 1993, pp. 61-73



Modelling and Simulation

Edited by Giuseppe Petrone and Giuliano Cammarata

ISBN 978-3-902613-25-7

Hard cover, 688 pages

Publisher I-Tech Education and Publishing

Published online 01, June, 2008

Published in print edition June, 2008

This book collects original and innovative research studies concerning modeling and simulation of physical systems in a very wide range of applications, encompassing micro-electro-mechanical systems, measurement instrumentations, catalytic reactors, biomechanical applications, biological and chemical sensors, magnetosensitive materials, silicon photonic devices, electronic devices, optical fibers, electro-microfluidic systems, composite materials, fuel cells, indoor air-conditioning systems, active magnetic levitation systems and more. Some of the most recent numerical techniques, as well as some of the software among the most accurate and sophisticated in treating complex systems, are applied in order to exhaustively contribute in knowledge advances.

InTech Europe
University Campus STeP Ri
Slavka Krautzeka 83/A
51000 Rijeka, Croatia
Phone: +385 (51) 770 447
Fax: +385 (51) 686 166
www.intechopen.com

InTech China
Unit 405, Office Block, Hotel Equatorial Shanghai
No.65, Yan An Road (West), Shanghai, 200040, China
中国上海市延安西路65号上海国际贵都大饭店办公楼405单元
Phone: +86-21-62489820
Fax: +86-21-62489821